

OPTIMIZATION OF MOLDING PARAMETER EFFECT TO WARPAGE OF CAR
BUMPER BASED ON PLASTIC FLOW SIMULATION SOFTWARE

MOHD NASRUL MAHMUDI BIN MAHMUD

Report submitted in fulfillment of the requirements
for the award of the Bachelor of
Mechanical Engineering with Automotive Engineering

Faculty of Mechanical Engineering
UNIVERSITI MALAYSIA PAHANG

NOVEMBER 2009

ABSTRACT

This research is specially focused on finding optimum molding parameter effect to warpage of a car bumper. The objectives of this research are to study the influence of injection molding parameters on a car bumper and analysis model car bumper using plastic flow simulation software. The idea of this project came when injection molding is the most widely used process in automotive industry. Since the qualities of injection molded plastic part are mostly influenced by process condition, how to determine the optimum process conditions becomes the key to improving the part quality. The optimization of molding parameter effect is very important for the automotive industry because it gives beneficial effects in production costs due to less material being used and shorter cycle times. In this project, plastic flow simulation method is proposed to optimize the molding parameter effect to warpage of a car bumper. The car bumper will be analyzing using three different parameters such as mold temperature, melt temperature, and injection pressure. This parameter will be changed to see the effect on car bumper. The result shows that the plastic flow simulation method is an effective tool for the process optimization of injection molding.

ABSTRAK

Penyelidikan ini adalah memfokuskan kepada pencarian kesan parameter injection molding yang paling optimum ke atas bumper kereta. Objektif kajian ini adalah untuk mengkaji pengaruh parameter injection molding ke atas bumper kereta dan menganalisis model bumper kereta menggunakan perisian komputer 'Plastic Flow Simulation'. Idea projek ini bermula apabila proses injection molding ini banyak digunakan secara meluas dalam bidang industri automotif. Disebabkan kualiti bahagian plastik injection molding kebanyakannya dipengaruhi oleh keadaan proses, untuk mencari keadaan proses yang paling optimum menjadi kunci untuk meningkatkan lagi kualiti bahagian itu. Kesan parameter molding yang paling optimum adalah sangat penting dalam industri automotif kerana ia memberi lebih keuntungan dalam kos pengeluaran disebabkan oleh kurangnya bahan yang digunakan dan penggunaan masa yang lebih singkat. Dalam tesis ini, kaedah simulasi 'plastic flow' digunakan untuk mengoptimumkan kesan parameter molding ke atas bumper kereta. Bumper kereta itu akan dianalisis menggunakan tiga parameter yang berbeza seperti suhu acuan, suhu leburan plastic, dan tekanan suntikan. Parameter-parameter ini akan diubah untuk melihat kesan keatas bumper kereta. Keputusan menunjukkan kaedah simulasi 'plastic flow' ini adalah sangat berkesan untuk mengoptimumkan proses injection molding.

TABLE OF CONTENTS

	Page
SUPERVISOR’S DECLARATION	i
STUDENT’S DECLARATION	ii
ACKNOWLEDGEMENTS	iv
ABSTRACT	v
ABSTRAK	vi
TABLE OF CONTENTS	vii
LIST OF FIGURES	x

CHAPTER 1 INTRODUCTION

1.1	Introduction	1
1.2	Background	2
1.3	Problem Statement	3
1.4	Objectives of the Study	3
1.5	Scopes and Limitation	3
1.6	Organization of the Thesis	4

CHAPTER 2 LITERATURE REVIEW

2.1	Injection Molding	5
2.2	Machine Component	6
	2.2.1 Injection System	6
2.3	Theory of Injection Molding	8
	2.3.1 The Operation of Injection Molding	8
2.4	Parameter of Molding Process	9
	2.4.2.1 Identifying the Parameters	9
	2.4.2.2 Temperature	10
	2.4.2.1 Melt Temperature Control	10
	2.4.2.2 Mold Temperature Control	12
2.5	Pressure	13
	2.5.1 Injection Unit	13
	2.5.2 Initial Injection Pressure	13
	2.5.3 Holding Pressure (Secondary Pressure)	14
	2.5.4 Back Pressure	15
2.6	Optimization Methodology	15
	2.6.1 Causes of Warpage	16
2.7	2.6.2 Classifying the cause of Warpage	16
	Moldflow	19
	2.7.1 Moldflow Plastic Insight (MPI)	20

CHAPTER 3 METHODOLOGY

3.1	Injection Molding	22
3.2	Flow Chart	23
3.3	Part Modeling	24
3.4	Material Selection	24
3.5	Plastic Flow Analysis	25
	3.5.1 Analysis Sequence	25
	3.5.2 Analysis	26

CHAPTER 4 RESULTS AND DISCUSSION

4.1	Introduction	27
4.2	SolidWorks Software	27
4.3	Melt Temperature Analysis	28
4.4	Mold Temperature Analysis	34
4.5	Injection Pressure Analysis	39

CHAPTER 5 CONCLUSION AND RECOMMENDATIONS

5.1	Conclusions	46
5.2	Recommendation	47

REFERENCES	49
-------------------	----

APPENDICES

A	Melt Temperature	50
B	Mold Temperature	59
C	Injection Pressure	68
D	SolidWorks Proton Saga Car Bumper	77

LIST OF FIGURES

Figure No.	Title	Page
2.1	A single screw injection molding machine for thermoplastics	6
2.2	A reciprocating screw	7
2.3	(a) Nozzle with barrel in processing position. (b) Nozzle with barrel backed out for purging.	7
2.4	Schematic of the injection molding process	9
2.5	Main processing parameters	10
2.6	Heating cylinder	11
2.7	Initial injection pressure	13
2.8	Holding pressure	14
2.9	Back pressure	47
2.10	Differential shrinkage for both unfilled and filled materials	17
2.11	Part warpage due to (a) non-uniform cooling in the part, and (b) asymmetric cooling across the part thickness	18

2.12	Larger volumetric shrinkage due to the high crystallization level in the slow cooling areas leads to differential shrinkage and thus part warpage	18
2.13	The poor cooling of the mold wall on the ribbed side causes a slower cooling of the material on that one side, which can lead to part warpage	18
3.1	Project flow chart	24
4.1	Fill time for different melt temperatures (a) 230°C (b) 240°C (c) 250°C	28
4.2	Air traps for different melt temperatures (a) 230°C (b) 240°C (c) 250°C	29
4.3	Volumetric shrinkage for melt different temperatures (a) 230°C (b) 240°C (c) 250°C	30
4.4	Weld lines for different melt temperatures (a) 230°C (b) 240°C (c) 250°C	31
4.5	Deflection for different melt temperatures (a) 230°C (b) 240°C (c) 250°C	32

4.6	Fill time for different mold temperatures (a) 40°C (b) 50°C (c) 60°C	34
4.7	Air traps for different mold temperatures (a) 40°C (b) 50°C (c) 60°C	35
4.8	Volumetric shrinkage for different mold temperatures (a) 40°C (b) 50°C (c) 60°C	36
4.9	Weld lines for different mold temperatures (a) 40°C (b) 50°C (c) 60°C	37
4.10	Deflection for different mold temperatures (a) 40°C (b) 50°C (c) 60°C	38
4.11	Fill time for different injection pressure (a) 20 MPa (b) 30MPa (c) 40MPa	40
4.12	Air traps for different mold temperatures (a) 30 MPa (b) 40 MPa (c) 50 MPa	41
4.13	Volumetric shrinkage for different mold temperatures (a) 18 MPa (b) 20MPa (c) 22 MPa	42
4.14	Weld lines for different mold temperatures (a) 18 MPa (b) 20 MPa (c) 22 MPa	43

4.15	Deflection for different mold temperatures (a) 18 MPa (b) 20 MPa (c) 22 MPa	44
------	--	----

CHAPTER 1

INTRODUCTION

1.1 Introduction

Injection molding is the most common method of production, especially in car manufacturing from the smallest component to entire body panels of car. Injection molding can also be used to manufacture parts from aluminum or brass (die casting). The melting points of these metals are much higher than those of plastics. This makes for substantially shorter mold lifetimes despite the use of specialized steels. Nonetheless, the costs compare quite favorably to sand casting, particularly for smaller parts. Hence, majority of injection molding is applied to thermoplastic polymers. The process started from heating thermoplastic material until it melts and forced into a steel mold. It cools down and solidifies before part is ejected.

There are numerous variables affect the injection-molding process. In fact, a recent study itemized more than 200 different parameters that had a direct or indirect effect on the process. It was found that the parameter can be categorized in three major divisions: temperature, pressure, and time. An adjustment to any one of these has a direct influence on some, or all of the other parameter. For a minimum-defect, high quality production, it is critical to control as many parameters as possible. The more

parameters that are controlled, the higher the quality level of the products being molded. [6]

Many companies always strive to produce high-quality parts while lowering their costs; however, significant time delays and increasing costs may occur if part design is not carefully evaluated or the injection process is not completely understood. Injection molding process allowed success in the production of newer material, especially for complex shape parts. Using injection molding technology, various parts can be made in shorter cycle times. Therefore, optimization of molding parameter effect is very important for the industry.

The process of injection molding offers many advantages which are unattainable using other methods. Over the years, injection molding technology has developed into a standard and popular machining technology. In many operations, especially in automotive industry, the injection molding process was chosen because of high quality part surfaces, good mechanical properties, low cost, and light weight. It also gives beneficial effects in production costs due to less material being used and shorter cycle times.

1.2 Background

In this study, the optimization of molding parameter effect to warpage of car bumper was investigated using plastic flow simulation software. The simulation was conducted using different parameters like mold temperature, melt temperature, and injection pressure. The objective of the process optimization is to select the optimal control variables in injection molding under certain given constraints, in order to obtain the best part quality. It is found that different values of parameter give different result of

finished molded product. A car bumper was modeled using CAD software before being analyzed using plastic flow simulation software.

1.3 Problem statement

The objective of finding optimum molding parameter is to achieve high quality products, cut product development, cycle times, and cost. Previously, production engineers used trial-and-error method to determine optimal process parameter setting for plastic injection molding. However, this method is unsuitable in present plastic injection molding because the increasing complexity of product design and the requirement of multi-response quality characteristics. However, the selection of parameter is still not completed. For every parameter that had found, another would appear. Even items such as humidity and ambient temperature had an effect on the molding process. Although there are so many different variables, it is not impossible to get control of the injection-molding process. What is needed is a more practical approach to understanding all these parameters, and targeting those that have the greatest effect on the overall quality and cost-effectiveness of finished molded product.

1.4 Objectives of the study

The objectives of this thesis are as follows:-

1. Study the influence of injection molding parameters effect to warpage of a car bumper.
2. Analyze model car bumper using plastic flow simulation software.

1.5 Scope and limitation

1. A proton saga car bumper was modeled using CAD software.
2. The model car bumper will be analyzed using plastic flow simulation software.
3. The parameter of injection molding process were selected such as mold temperature, melt temperature, and injection pressure.
4. Study the effect of warpage in injection-molded part.

1.6 Organization of the thesis

This thesis consists of five chapters. Chapter 1 is the introduction about this study. Chapter 2 is the review of literature which discusses the methods and findings previously done by the other people which are related to the study. Chapter 3 is methodology which explains the approaches and methods used in performing the thesis. Chapter 4 is the chapter which reports the outcomes or results and discussion from the project and chapter 5 consists of the recommendation and conclusion.

CHAPTER 2

LITERATURE REVIEW

2.1 Injection molding

Injection molding is one of the most exploited industrial processes in the production of plastic parts. Its success relies on the high capability to produce 3D shapes at higher rates. The part of injection molding machines are mold clamp, ejector pins, moving die, stationary die, barrel hopper and motor. The pellets or granules are fed into the heated cylinder, and melt is forced into the mold either by a hydraulic plunger or by the rotating screw system of an extruder. As in plastic extrusion, the barrel is heated externally to promote melting of the polymer. In injection molding machines, however, a far greater portion of the heat transferred to the polymer is due to frictional heating.

Modern machines are of the reciprocating or plasticating screw type with the sequence of operation. As the pressure builds up at the mold entrance, the rotating screw begins to move backward under pressure to a predetermined distance. This movement controls the volume of material to be injected. The screw then stops rotating and is pushed forward hydraulically, forcing the molten plastic into the mold cavity. The pressure developed usually range from 70 to 200 MPa.

For thermoplastic, the molds are kept relatively cool at about 90°C. Thermoset part are molded in heated mold at about 200°C, where polymerization and cross-linking take place.

After the part has cooled sufficiently (for thermoplastic) or cured (for thermoset), the molds are opened and the part is removed from the mold using ejector. Elastomers also are injection molded into discrete products by this processes. Because the material is molten when injected into the mold, complex shapes with good dimensional accuracy can be obtained.

2.2 Machine components

2.2.1 Injection system

The injection system consists of a hopper, a reciprocating screw and barrel assembly, and an injection nozzle, as shown in Figure 2.1. This system confines and transports the plastic as it progresses through the feeding, compressing, degassing, melting, injection, and packing stages.

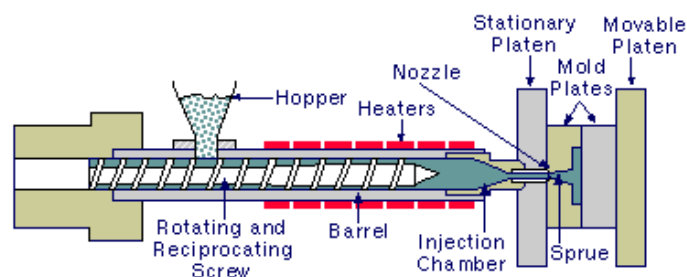


FIGURE 2.1: A single screw injection molding machine for thermoplastics.

- The hopper

Thermoplastic material is supplied to molders in the form of small pellets. The hopper

on the injection molding machine holds these pellets. The pellets are gravity-fed from the hopper through the hopper throat into the barrel and screw assembly. [9]

- The barrel

As shown in Figure 2.1, the barrel of the injection molding machine supports the reciprocating plasticizing screw. It is heated by the electric heater bands. [9]

- The reciprocating screw

The reciprocating screw is used to compress, melt, and convey the material. The reciprocating screw consists of three zones (illustrated below):

1. the feeding zone
2. the compressing (or transition) zone
3. the metering zone

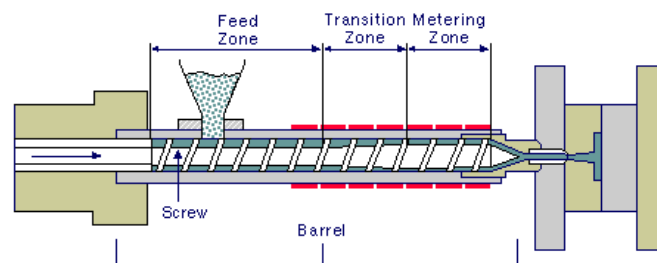


FIGURE 2.2: A reciprocating screw

- The nozzle

The nozzle connects the barrel to the sprue bushing of the mold and forms a seal between the barrel and the mold. The temperature of the nozzle should be set to the material's melt temperature or just below it, depending on the recommendation of the material supplier. When the barrel is in its full forward processing position, the radius of the nozzle should nest and seal in the concave radius in the sprue bushing with a locating ring. During purging of the barrel, the barrel backs out from the sprue, so the purging compound can free fall from the nozzle. These two barrel positions are illustrated below [9]:

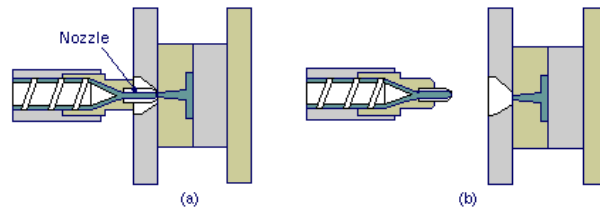


FIGURE 2.3: (a) Nozzle with barrel in processing position. (b) Nozzle with barrel backed out for purging.

2.3 Theory of injection molding

The theory of injection molding can be reduced to four simple individual steps: Plasticizing, Injection, Chilling, and Ejection. Each of those steps is distinct from the others and correct control of each is essential to the success of the total process. [8]

1. Plasticizing - describes the conversion of the polymer material from its normal hard granular form at room temperatures, to the liquid consistency necessary for injection at its correct melt temperature.
2. Injection - is the stage during which this melt is introduced into a mold to completely fill a cavity or cavities.
3. Chilling - is the action of removing heat from the melt to convert it from a liquid consistency back to its original rigid state. As the material cools, it also shrinks.
4. Ejection - is the removal of the cooled, molded part from the mold cavity and from any cores or inserts.

Repetition of these basic steps in sequence is the process of injection

2.3.1 The operations of an injection molding are:

- 1) Closing the mold: the mold closes so the cycle can begin.
- 2) Plasticizing the resin: (Figure 2.4a) the hopper feeds solid pellets or grains of the plastic resin into the barrel where it becomes molten due to the heating bands and

friction caused by the rotating screw. The molten plastic accumulates at the front of the barrel (the nozzle side) as the screw retracts to the rear of the barrel.

3) Injecting the resin: (Figure 2.4b) when enough molten plastic has accumulated for a full shot, a valve in the nozzle is opened and the screw rapidly advances forward, quickly injecting the plastic into the mold cavity.

4) Cooling the part: (Figure 2.4c) The screw continues to push plastic into the mold in order to create a holding pressure. This ensures adequate filling. As this happens, the molten plastic begins to cool and solidify toward the inside. This natural cooling is expedited by convection due to coolant flowing through channels inside the mold.

5) Ejecting the part: (Figure 2.4d) After adequate cooling time has elapsed, the mold is opened. Some sort of ejector device is actuated in this process and the part is ejected from the mold and collected. After this, the cycle repeats from step 1.

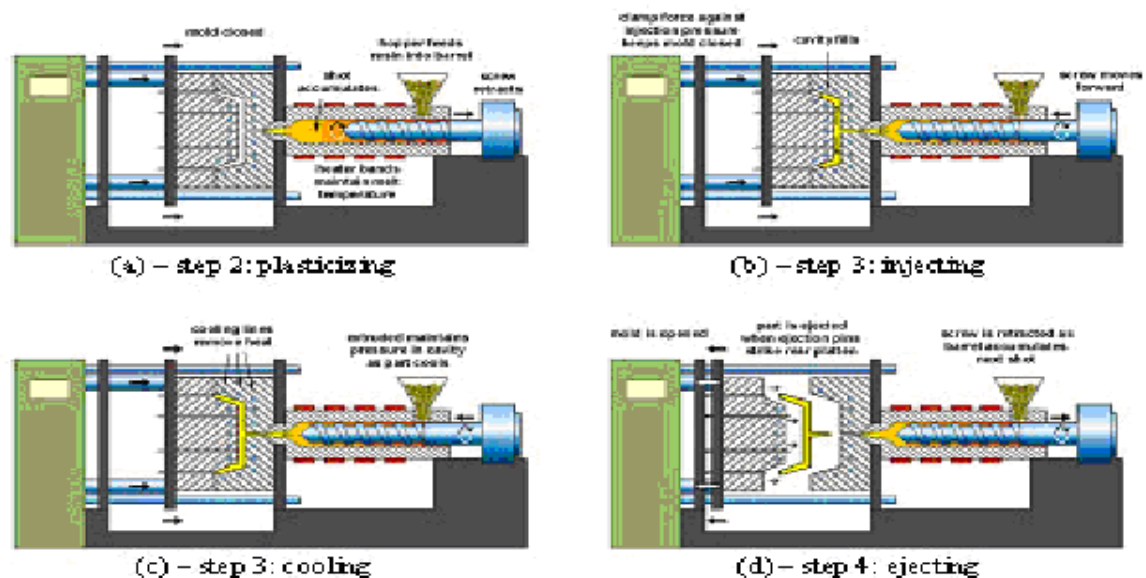


Figure 2.4: Schematic of the injection molding process

2.4 Parameter of the molding process

2.4.1 Identifying the parameters

There are numerous variable affect the injection molding process. For instance, injection pressure consisted of more than one item. There are initial injection pressures, second and up to fifth stage injection pressure, holding pressure, back pressure, and line pressure. All of these give direct effect on each other. Humidity and ambient temperature also have effect on the molding process. Shift changes, relief operators, fan blowing, housekeeping, age of equipment, size of machine, location of press, pressure of cooling water, all seemed to have direct or indirect effect on the injection molding process.

Figure 2.5 shows that all of the parameters involved can be placed into two main categories: temperature and pressure. All these relative importance of the categories is shown by size of the circles. Thus, temperature is the most important, followed by pressure. However, each is dependent on the other, and changing one will affect one or all of the others. [6]

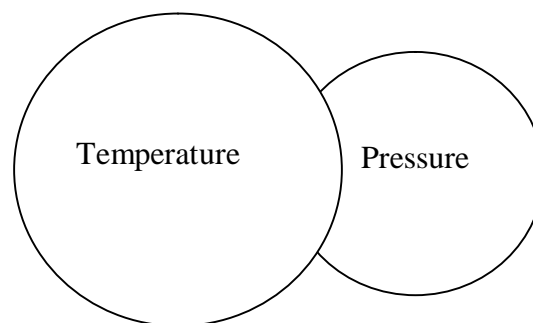


Figure 2.5: Main processing parameters

2.4.2 Temperature

A variety of temperatures affect the injection molding process, ranging from melt temperature to mold temperature, and including even ambient temperature.

2.4.2.1 Melt Temperature Control

Melt temperature is the temperature at which the plastic material is maintained throughout the flow path. This path begins where the plastic material is transferred from the machine hopper into the cylinder of the injection unit. Then the material is augered through the heating cylinder and into the machine nozzle. From there it is injected into the mold, where it must travel along a runner system (if one exists), through the gates, and into the cavities that are machined into the mold. The temperature of the melt must be controlled along the path, starting with the heating cylinder. [6]

Figure 2.6 shows the heating cylinder is wrapped with heater bands. These are electrical heaters shaped like hinged bracelets that mount around the outside of the heating cylinder. There are three main heating zones to the heating cylinder: the rear zone, the center zone, and the front zone. In addition, there is usually at least one heater band fastened around the machine nozzle, an area referred to as the nozzle zone. [6]

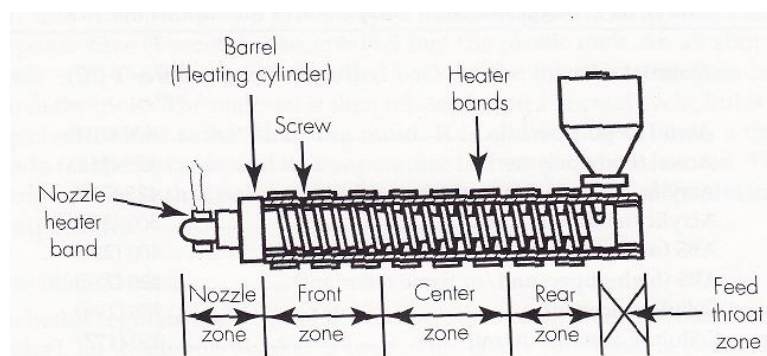


Figure 2.6: Heating cylinder

The plastic for the injection process should be brought up to proper temperature gradually as the material drops from the hopper into the rear zone, where the initial heat begins to soften the material. Then the material is augered forward by the screw into the

center zone where the temperature is generally 10 to 20°F (5.6 to 11°C) higher than the in the rear zone. As the material travels to the front zone, the temperature is gain increased by 10 to 20°F, and finally the material is ready to be injected into the mold. It is held at this point until the previous molding cycle is complete, at which time the mold opens, parts are ejected, the mold closes, and the next cycle begins. The charge of preheated plastic is then injected into the mold. [6]

Besides absorbing heat from the externally mounted heater bands, the plastic material absorbs a large amount of heat from the friction caused by the augering action of the injection screw. The screw rotates to bring fresh material into the heating cylinder and prepare it for the coming cycle. While being pulled along, the new material is squeezed between the flights of the screw and the inside wall of the injection barrel. The friction generates heat, which is absorbed by the plastic. [6]

The main point here is that plastic must be heated to the proper temperature for injection. Melt temperature is measured at the nozzle as the plastic exit the machine, before it enters the mold. It is measured by taking an “air shot” and plunging a probe from a measuring instrument with a fast response time (1 second id acceptable) into the plastic melt. An air shot is made with the injection sled pulled back so the injection unit does not touch the mold. The material is then released as in a normal cycle, but it is injected into air rather than the mold. It is allowed to fall onto a tray made for the purpose and its temperature is then quickly measured. The temperature at that point should be within 10°F (5.6°C) of the desired temperature. [6]

2.4.2.2 Mold Temperature Control

The plastic material is now ready to flow into the mold. First, it must travel through the machine nozzle, which is the last heating zone provided by the machine. After the material exits the nozzle and enters the mold, it immediately begins to cool down as the mold absorbs heat from it. The rate at which this heat is absorbed determines how far the plastic will flow before it begins to solidify and stop moving. Each product depending on its design and plastic material, demand specific cooling rates, and this rate of cooling is critical to product quality. Therefore, the mold

temperature must be regulated in order to regulate the cooling rate of the plastic. This is done by connecting the mold to a temperature control unit that normally utilizes water as medium. The water is circulated through the mold and held at a preset temperature by heating or cooling in cycles. [6]

Every combination of plastic and product has a specific temperature at which the mold should be maintained to ensure quality molding. The mold temperature is measured directly from the molding surface of the tool with a solid probe on a pyrometer device. Usually, readings from several areas are averaged. The object of the cooling process is to lower the temperature of the molded plastic to the point at which it solidifies again. When the plastic reaches that point, it can be ejected from the mold with relative structural safety. That simply means that the plastic part will not move excessively, causing warpage, twisting, or other shrinkage related problem as the plastic continues to cool. [6]

2.5 Pressure

There are two areas in the injection machine that required pressure and pressure control: the injection unit and the clamp unit. They are closely related in that they are opposing pressure – the clamp unit must develop enough clamp pressure to overcome the pressure developed by the injection unit during the molding process.

2.5.1 Injection unit

Three basic types of pressure are developed by the injection unit: initial pressure, hold pressure, and back pressure.

2.5.2 Initial Injection Pressure

This is the first pressure that applied to the molten plastic. It develops as result of main system hydraulic pressure pushing against the back end of the injection screw (or plunger) (figure 2.7)

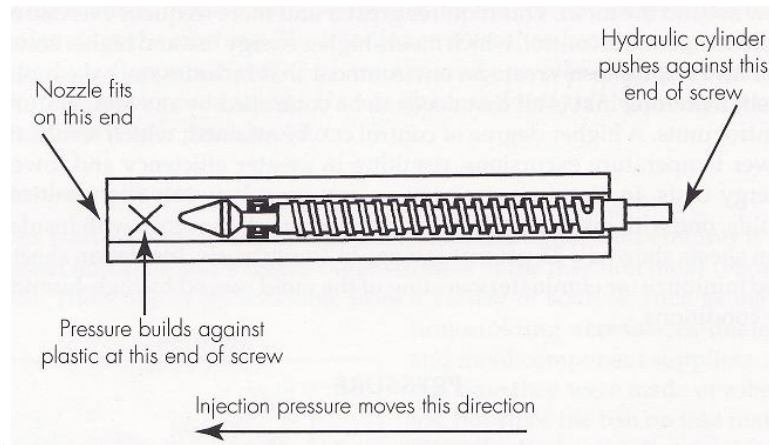


Figure 2.7: Initial injection pressure

The amount of pressure developed by the main system is on the order of 2000 psi (13,789 kPa). Some systems are capable of producing more than that, but 2000 psi is the most common line pressure. This pressure is converted to a maximum of 20,000 psi (137,890 kPa) at the nozzle of the injection unit (where the plastic first enters the mold) by the design and shape of the injection screw. In most cases, the full 20,000 psi is not required for filling a mold, and most products can be molded in a range of from 5000 to 15,000 psi (34,472 to 103,418 kPa). The pressure actually required depends on the plastic being molded, the viscosity and flow rate of the plastic, and the temperatures of the plastic and the mold. [6]

To summarize, initial injection pressure is used to create the initial filling of the mold. It should be set at the highest practical value to fill the mold with the fastest practical speed.